FEM STRUCTURAL ANALYSIS OF A HELICAL DRILL BY USING CATIA

Nioaţă Alin, *lecturer PhD. eng., "Constantin Brâncuşi" University of Târgu-Jiu, Romania*

ABSTRACT: With the aid of the finite element analysis method can be studied problems whose complexity is given by the complicated geometric configuration of the bodies, material inhomogeneities, anisotropy of the materials, composite materials, etc. These problems occur frequently in practice at the various stages of development of a product, even when a product already exists, but the problem of improving its characteristics is raised.

KEY WORDS: CATIA, cutting tool, helical drill , force.

1. INTRODUCTION

CATIA (Computer Aided Three Dimensional Interactive Applications) is a product of Dassault Systemes company, one of the most advanced integrated CAD/CAM/CAE platforms based on the latest technologies in the field of informatic industry [2],[3],[4].

Basic functionalities [2],[3],[4]:

- advanced design of mechanical parts;
- interactive producing of the assemblies;
- automatic obtaining of projections of the current piece or ensemble;
- the possibility of designing in a parameterized way;
- allows designing the parts and assemblies directly in three dimensions, without first drawing the sketches in two-dimensional representation;
- uses virtual prototypes that allow:
  - -testing of resistance to various stresses;
  - - verifying whether an assembly is dismountable or not;
  - -verifying if the mobility of components, relative to each other, does not generate collisions;
  - - has modular structure.

Although the number of modules is very large, a few can be considered as basic [2],[3],[4]:

- **CATIA Sketcher** - creates a sketch of a profile in two dimensions;

- **CATIA Part Design** –is used for the construction of mechanical parts in three dimensions;
- **CATIA Assembly Design** - generates a set of parts using various mechanical constraints for positioning them and for determining surface contacts;
-**CATIA Drafting** - allows obtaining the work drawings of the parts and assemblies created;
- **CATIA Knowledge Advisor** - allows parameterized design using specific tools such as formulas, parameters, rules and reactions, activated only after a pre-established condition has been met;
- **CATIA Composites Design** - allows the creation of composite assemblies;
- **CATIA Analysis & Simulation** - allows finite element analysis of parts / assemblies;
- **CATIA Machining** - allows designing and simulation of operations on numerically controlled machines;

2. MODELLING OF THE PIECE

In this stage it is considered a helical drill with a diameter of φ16 (figure 1) with the following characteristics:

- angle of point: 2χ=116º-120º;
- angle of dip of the helical grooves: ω=28º-29º;
- seat angle: α= 10º-12º.

The drill is made of Rp3 fast steel with the following international equivalence (Table 1):

Table 1. Rp3 symbolization and its international equivalence

SR	JIS	GOST	Werkstoff	Bohler	AISI/SAE	UNE
Rp3	SKH2	R18	W.1.3355	S200	T1	F5520 18-0-1

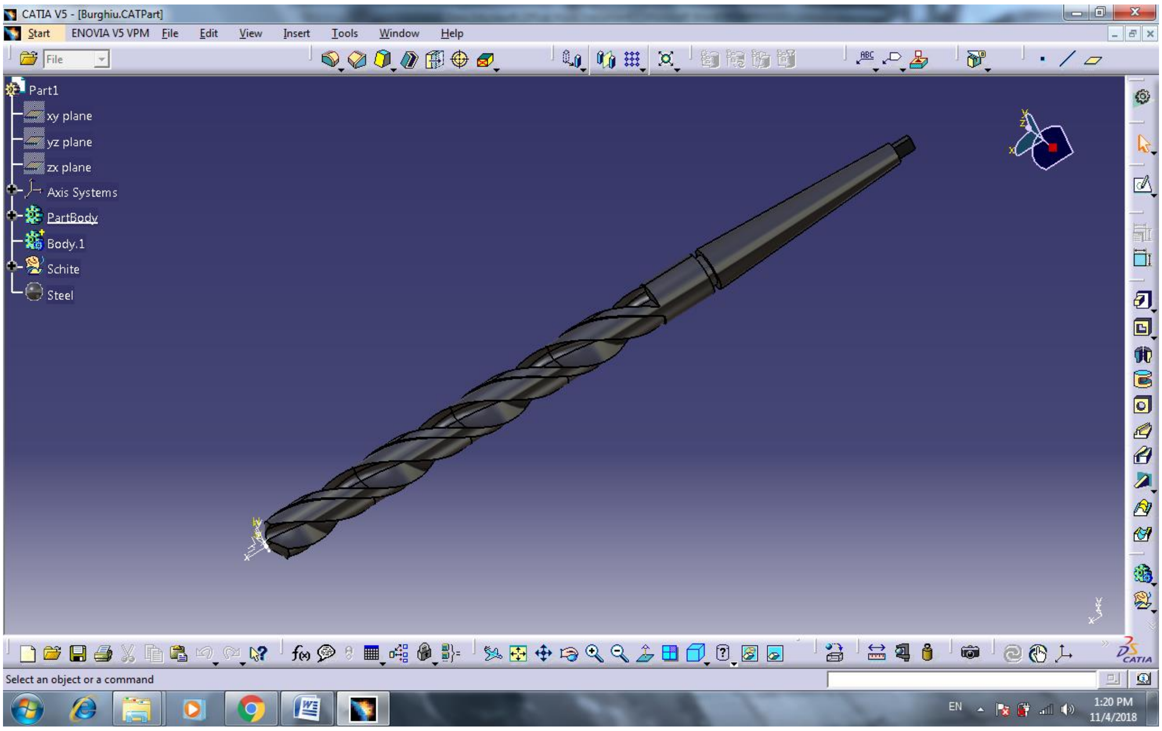

Figure 1. Helical drill

3. FEM Analysis

After solid modelling in the CATIA Part Design module, the piece is considered to be made of a material (steel) having the following physical and mechanical properties, important during analysis: Young's module ($2x10^{11}N/mm^2$), Poisson's coefficient (0.266) , density (7860 kg/m^3), coefficient of thermal expansion ($1,17x10^{-5}$ ºK), permissible strength ($2,5x10^8$ N/m^2) [4],[5].

The CATIA Generative Structural Analysis module is accessed from the Start - Analysis & Simulation menu and the Static Case type is determined , the specification shaft simultaneously displaying the element bearing the same name . Although the CATIA program defines the network of nodes and elements(discretization), it is recommended to edit this and determine the size of the finite element, the maximum tolerance between the discretized model and the real model used in the analysis (Absolute sag), the type of element(Element type), and so on [1],[2],[3]..

Next, a Clamp restraint (Figure 2) is applied to the gripping surface.

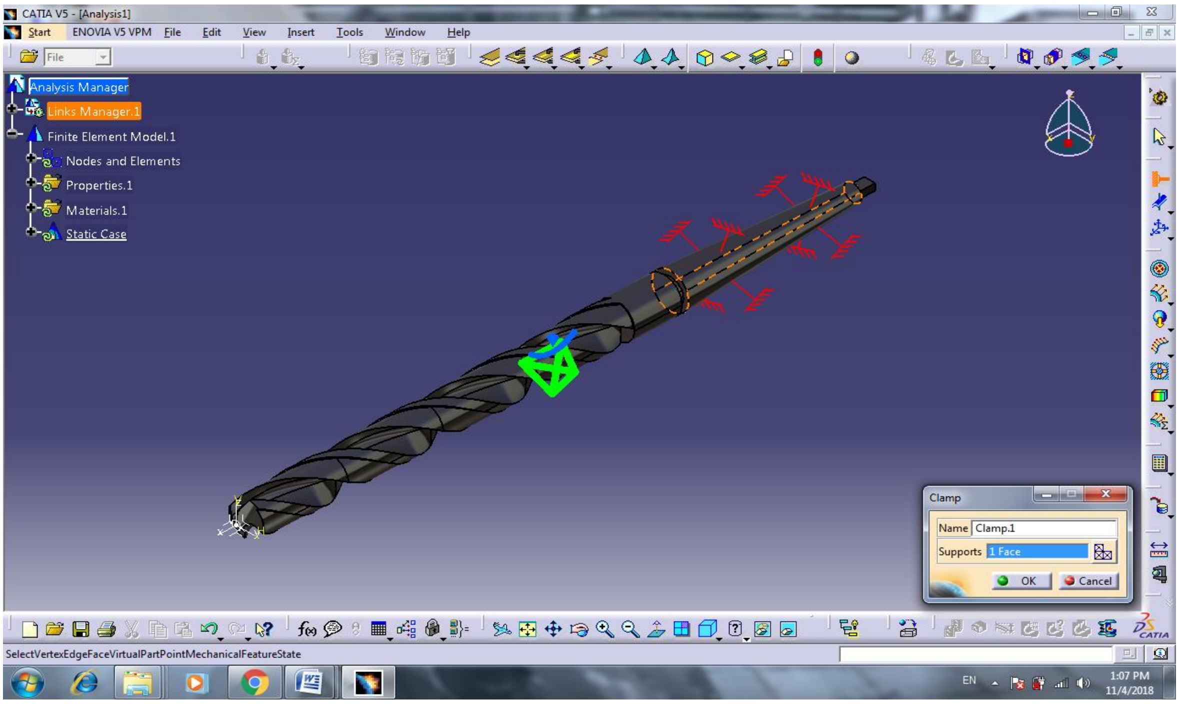

Figure 2. Applying supports

On the two active edges of the helical drill, the cutting forces resulting from the machining process are applied, having the values F_x = 100N, F_y = 500N şi F_z = 100N. In the specification shaft, elemental Distributed Force 1 becomes available, the force being specified by the arrows on the surface (Figure 3,4).

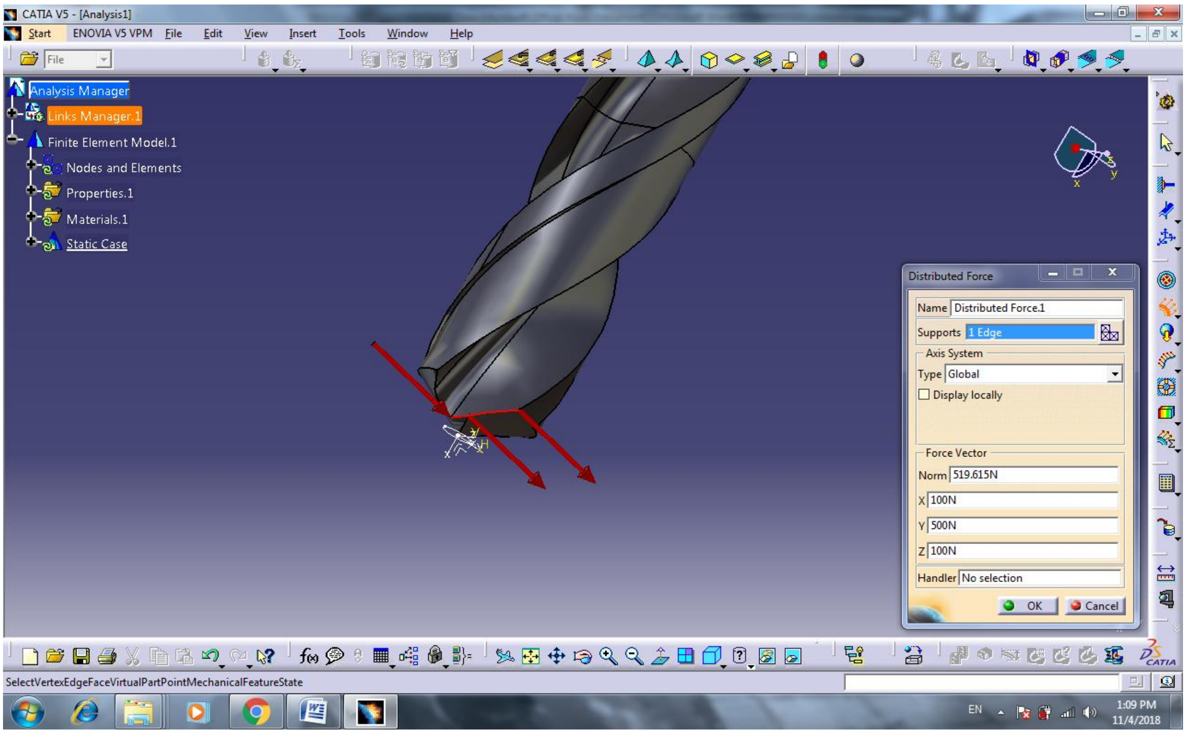

Figure 3. Applying forces - first on an edge

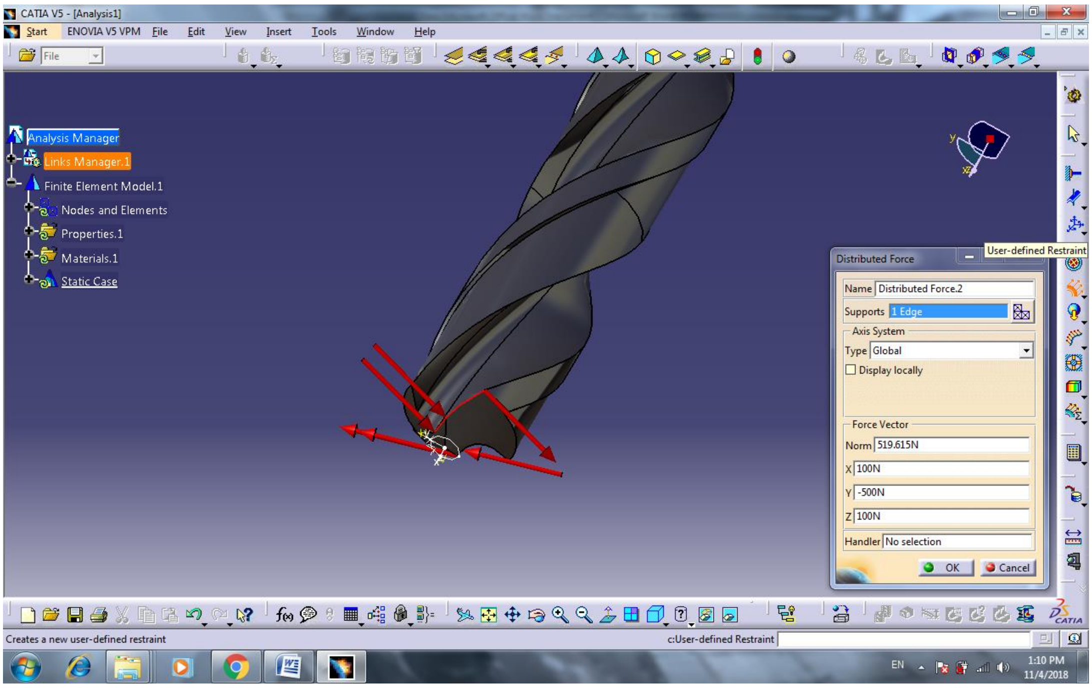

Figure 4. Applying forces on the second edge

Once the restrictions and loading have been established, the actual step of the calculation (analysis) follows. By clicking the Compute icon on the toolbar All option is selected , the first effect of the action being to update the Static Case Solution (Figure 5). Once the calculation is complete, the user has the Image bar tools available to view the results. The specifications tree is completed according to the inserted images. Figure 6 shows the image (using Von Mises Stress, Deformation, Principal Stress and Precision) corresponding to the calculation of the piece model and load considered, specifying that the deformations are graphically presented slightly overstated in order to ease the stage of determining the conclusions of the analysis [1],[5].

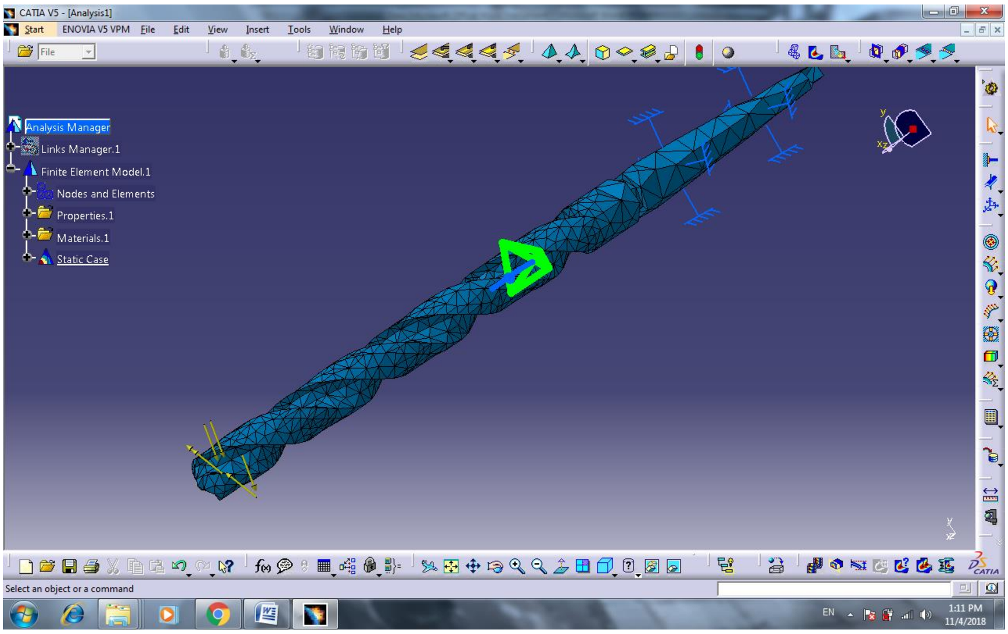

Figure 5. Discredited drill

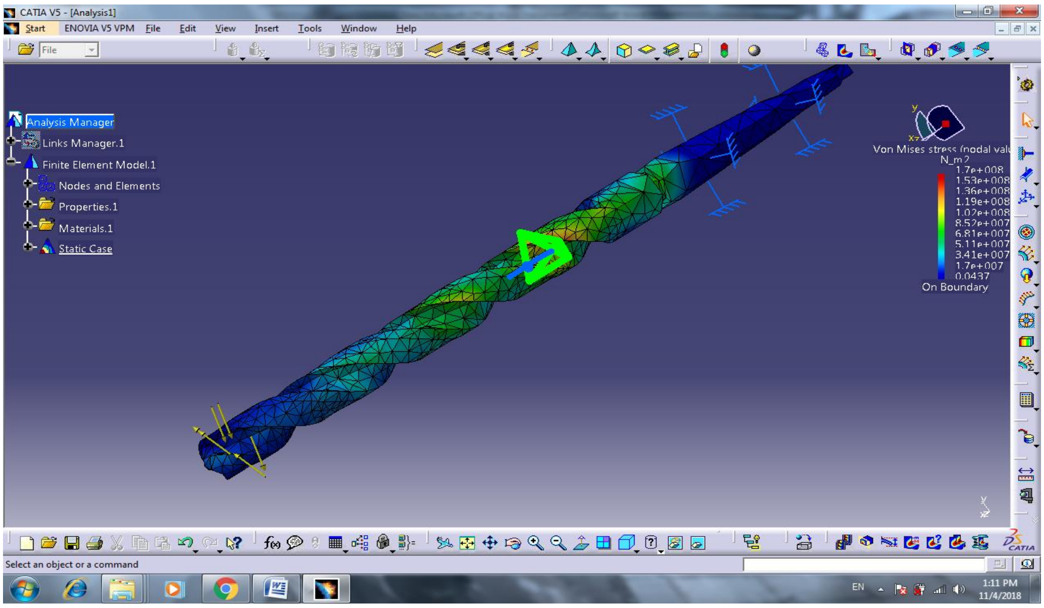

Figure 6. Discredited drill but viewing tense areas (red - strong, blue - weak)

In Figure 7, along with this window, the color palette accompanying the result-Von Mises image is also presented. The lowest stress values are at the bottom of the palette and the highest ones at the top of the palette. The dialog box also contains the explicit values in the Extreme Values area. The blue and light blue colors indicate low stresses, and the yellow to red colors are high ones.

Considering that the permissible strength of the material is 2.5 x 108 N/m^2, an initial conclusion can be drawn that the piece model will withstand the applied forces.

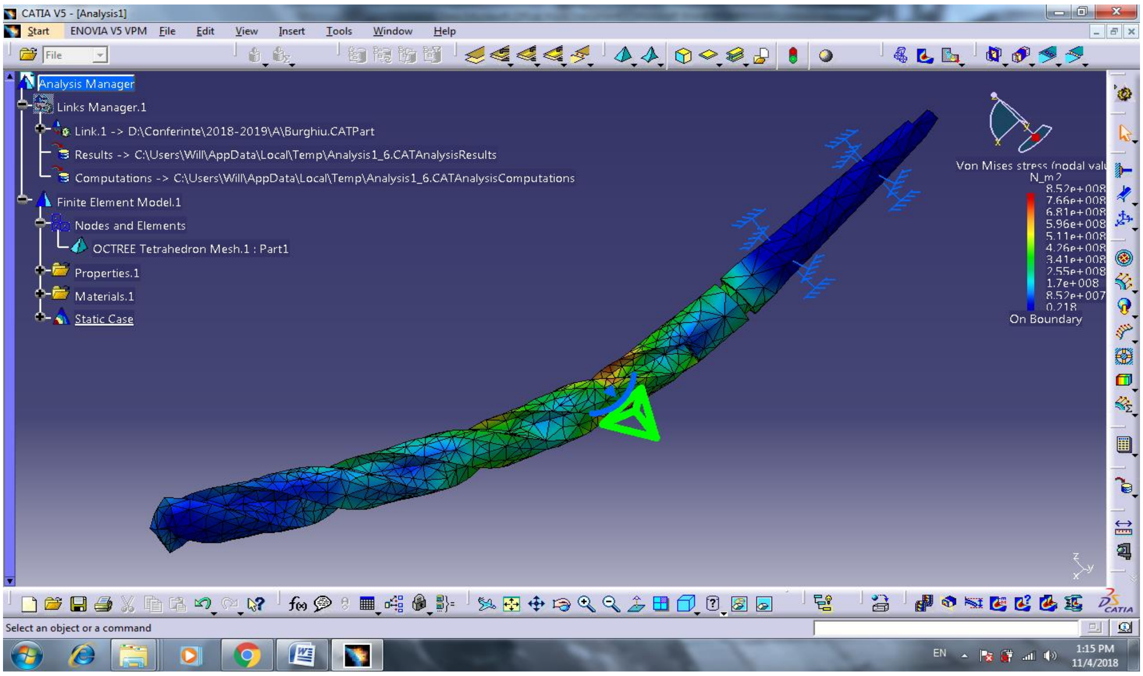

Figure 7. Drill deformed to the maximum deformation value

4.CONCLUSIONS

Finite element analysis using the CATIA program is a modern method for studying different contacts, allowing the determination of important parameters for the study of different contacts. In the case of hertz contacts, accurate information on the state of stresses at the contact level is obtained. The point contact analysis using the classical method (Hertz theory) did not allow such accurate results to be obtained. Von Mises stress determination allows a stress state analysis thus allowing for a correct quantification of this state and the possibility of identifying the deterioration that occurs.

BIBLIOGRAPHY

[1] **Ciofu Florin** - Hip implant analysis with CATIA, Revista Fiabilitate şi Durabilitate, Nr.2/2017, ISSN 1844-640X, pag.99.

[2] **Ghionea, I.G.** – CATIA v5 – culegere de aplicaţii pentru activităţi de laborator, format electronic, Universitatea Politehnică Bucureşti, 2015;

[3] **Ghionea, I.G.** – CATIA v5 – aplicaţii în inginerie mecanică, Editura BREN, Bucureşti, 2009;

[4] **Ghionea, I.G.** – Proiectare asistată în CATIA v5 – elemente teoretice şi aplicaţii, Editura BREN, Bucureşti, 2014;

[5] **Olariu, V., Brătianu, C.** - Modelare numerică cu elemente finite, Editura Tehnică, Bucureşti, 1986.